13th Australasian Fluid Mechanics Conference
Monash University, Melbourne, Australia
13-18 December 1998

NUMERICAL STUDY OF THE VORTEX ROLL-UP IN A SQUARE CAVITY USING A FRACTIONAL-STEP METHOD

C.K. TAN[1], A.S.H. OOI[2] and M.S. CHONG[1]

[1]Department of Mechanical and Manufacturing Engineering
Melbourne University, Parkville, Victoria, AUSTRALIA, 3052
[2]Center For Turbulence Research
Bldg. 500, Stanford, California 94305-3030, USA.

ABSTRACT

This paper presents preliminary results from a numerical simulation of a vortex roll-up in front of a square piston moving in a square duct. The numerical algorithm utilises the well-known fractional-step method to solve the full viscous incompressible Navier-Stokes equation. A staggered grid with 64×64 resolution and appropriate boundary conditions were used to simulate the flow inside a square cavity. The velocity vector fields, vorticity contours of the flow field and streamline patterns, along with other relevant plots are presented for flows with $Re = 2500$ and a brief analysis of these results is provided.

INTRODUCTION

This study has been motivated by the work of Allen (1996) who performed an extensive experimental study into the formation of vortex sheets close to the junction of moving surfaces and successfully extended the results of Tabaczynski *et al.* (1970). Tabaczynski *et al.* (1970) observed a vortex roll-up structure, using a circular piston to create a moving plane with respect to circular duct walls. Analytical solutions were then postulated which successfully described the collapse of the non-dimensionalised experimental data. A diagram of a typical vortex roll-up structure obtained by Allen (1996) is as shown in Figure 1.

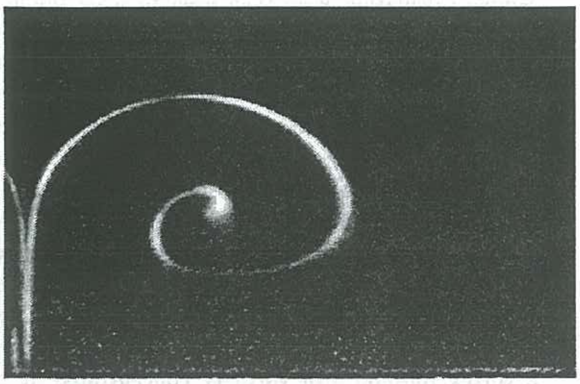

Figure 1: Vortex roll-up from experiments conducted by Allen (1996), $Re = 2446, t = 50s$

Allen (1996) extended these analytical solutions and classified three distinct regions of the vortex behaviour which are dependent on the Reynolds Number of the vortex.

A point vortex model was also developed by Allen (1996) which provided a locus of solutions which were found to be approximately similar to experimental data. A continuous vortex sheet model was then developed which assumed the sheet scales in a self-similarity behaviour. This allowed the utilisation of the Birkhoff-Rott equations to be transformed into an integro-differential equation. This model successfully produced a vortex sheet shape similar to that observed from experiments.

This paper thus serves to introduce a numerical method to study the development of the vortex roll-up in front of a moving piston.

NUMERICAL ALGORITHM

It is a well known fact that the fractional step method is an efficient method to solve the time-dependent viscous incompressible Navier-Stokes equations. This method is used here to solve the full two-dimensional Navier-Stokes equations in primitive variable form to provide a numerical solution to the corner flow problem. The method used is based on the work introduced by Chorin (1969) and Kim *et al.* (1985), and utilises a staggered grid method which was introduced by Harlow *et al.* (1965) to represent a finite-difference representation of the Navier-Stokes equations onto a square cavity, with the appropriate boundary conditions.

The Navier-Stokes equation for two-dimensional flow is as shown below together with the continuity equation:

$$\begin{aligned}
\frac{\partial u}{\partial t} + \frac{\partial uu}{\partial x} + \frac{\partial uv}{\partial y} + \frac{\partial p}{\partial x} &= \frac{1}{Re}\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) \\
\frac{\partial v}{\partial t} + \frac{\partial uv}{\partial x} + \frac{\partial vv}{\partial y} + \frac{\partial p}{\partial y} &= \frac{1}{Re}\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right) \\
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} &= 0 \qquad (1)
\end{aligned}$$

Here, x and y are the spatial coordinates and u and v are the velocity components respectively, with Re being the Reynolds number and p being the pressure.

Rewriting Eq.(1) in vectorial form and applying the fractional-step method yields the following:

$$\vec{u}^{*} = \vec{F}^{n} \quad (2)$$
$$\vec{u}^{n+1} = \vec{u}^{*} - \Delta t \left[\nabla \left(P^{n+1}\right)\right] \quad (3)$$

with

$$D\left(\vec{u}^{n+1}\right) = 0 \quad (4)$$

where

$$\vec{F}^{n} = \vec{u}^{n} + \Delta t \left[-\left(\vec{u}^{n} \bullet \nabla\right)\vec{u}^{n} + \nu\nabla^{2}\vec{u}^{n}\right] \quad (5)$$

D is the divergence operator, ∇ is the gradient operator and P is a scalar proportional to pressure. $\vec{u}^{n+1}$ and $\vec{u}^{n}$ are velocities at times $t+\Delta t$ and t respectively while Δt is the incremental time step, which is set to 0.1 here. $\vec{u}^{*}$ is known as the intermediate velocity and can be computed from Eq.(5). This computation forms the first-half step of the fractional-step method.

By taking the divergence of Eq.(3) and utilising Eq.(4), the following can be obtained:

$$\nabla^{2}\left(P^{n+1}\right) = \frac{1}{\Delta t} D\left(\vec{u}^{*}\right) \quad (6)$$

This is effectively a discrete Poisson equation for pressure, and requires the incorporation of all velocity boundary conditions. By means of a transforms method, used by Kim *et al.* (1985), Eq.(6) can be solved for P^{n+1} by cosine-transforming both sides of Eq.(6) and solving for the tridiagonal system of equations using a simple inversion method of tridiagonal matrices. These values are then cosine-transformed back into physical space, providing the values of pressure at each node on the staggered-grid.

These pressure values are then substituted into Eq.(3) and the velocity of the flow at the next time step can be computed, forming the second-half step of the fractional-step method.

GRID DEPENDENCE OF SOLUTIONS

The accuracy of the numerical scheme was checked by examining the grid dependence of the solution. This was done by refining the grid to a 128×128 grid. No significant difference in the solutions were obtained suggesting that the solutions were independent of grid size.

Preliminary results of the 64×64 computation is presented here as work on the 128×128 computation in still continuing.

PRELIMINARY RESULTS

To simulate a piston moving from left to right through a square piston duct, the upper and lower boundaries were set to move in the left direction. The computed velocity vectors is as shown in Figure 2, while Figure 3 shows the calculated trajectory of the vortex core.

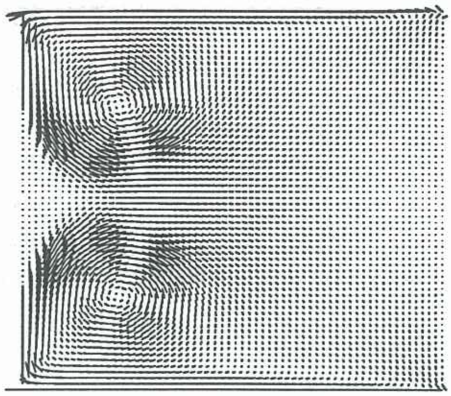

Figure 2: Velocity vectors of the simulated flow

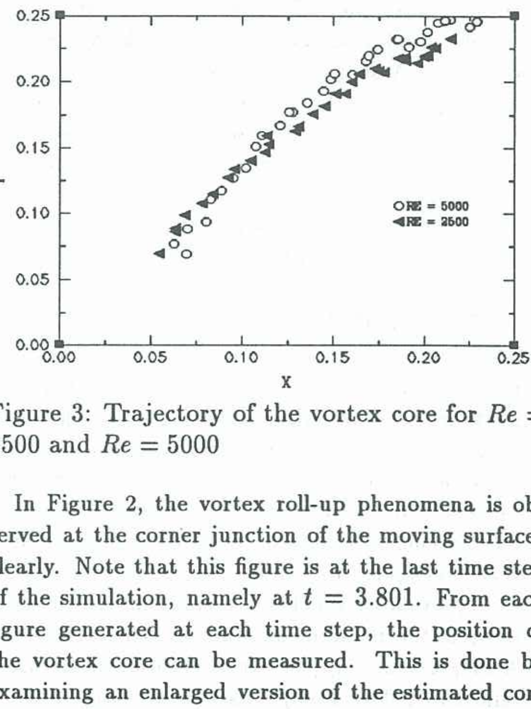

Figure 3: Trajectory of the vortex core for $Re =$ 2500 and $Re = 5000$

In Figure 2, the vortex roll-up phenomena is observed at the corner junction of the moving surfaces clearly. Note that this figure is at the last time step of the simulation, namely at $t = 3.801$. From each figure generated at each time step, the position of the vortex core can be measured. This is done by examining an enlarged version of the estimated core location of the vortex and approximating its position. Simple calculation were then used to scale the location of the core correctly, with respect to the scale of the diagram. Nevertheless, this method will require refinement in future works.

Figure 3 shows the trajectory of the vortex core for $Re = 2500$ and also superimposed is a later simulation for $Re = 5000$. These results show good agreement with experimental results reported by Allen (1996). Errors in determining the location of the core is approximately 1%, which is reasonable.

Figure 4 shows the contours of the vorticity of the flow field at the last time step, $t = 3.801$ and shows a vortex roll-up, with vorticity concentrated at the vortex core. Allen (1996) has mentioned that the center of vorticity would provide an accurate description

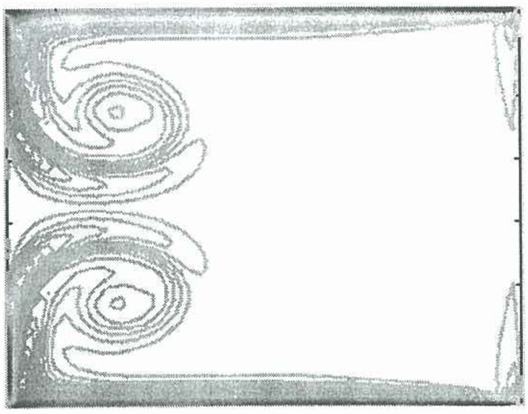

Figure 4: Vorticity contour of the flow field

of the position of the vortex core. However, a suitable experimental method to determine the region of maximum vorticity in a typical flow field is still being investigated.

Observations of the coloured version of Figure 4 clearly show the presence of secondary vorticity in the flow. This secondary vortex is located between the primary vortex system and the left boundary of the cavity. A clearer indication of this secondary eddy can be seen if a streamline pattern of the flow field is plotted.

A typical streamline pattern of the flow field is as shown in Figure 5 below:

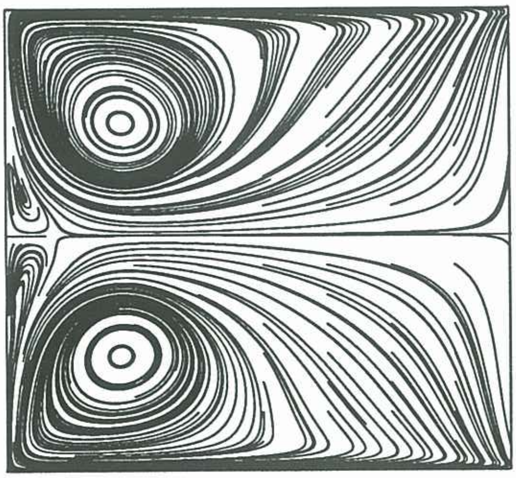

Figure 5: Streamline pattern of the flow field

Observations of Figure 5 again show a secondary vortex formation between the surface of the moving piston and the primary vortex roll-up. The strong tendency of the boundary layer on the wall of the duct to separate and roll-up into the vortical structure can also be clearly seen.

Figure 6 shows the velocity profile across the center of the cavity at the start and end of the simulation, namely at $t = 0.001$ and $t = 3.801$.

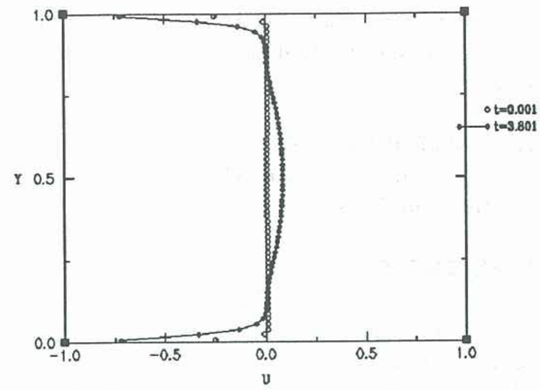

Figure 6: Velocity profile across the cavity at the first and last time step for $Re = 2500$

Burggraf (1966) suggested that the structure of flows in a primary eddy can be shown most clearly by a graph of the velocity profile across the eddy. This was done by examining the u velocity component at $x = L/2$, *i.e.* the middle of the cavity.

It can be observed that at $t = 0.001$, the velocity profile is approximately flat across the entire cavity with the exception of the two horizontal walls which is moving in the left hand direction. However, at the last time step $t = 0.3801$, a maximum velocity exists and is located at $y = L/2$. The point where the velocity is zero may be a good estimate of the y location of the core of the vortex. However, further investigation is required to verify this.

FUTURE WORK

From the preliminary results obtained, it can be noted that the numerical algorithm provides an alternative description of the flow at the corner junction of moving surfaces. Numerical simulations have shown the existence of a secondary vortex, located between the primary vortex and the moving piston wall.

Allen (1996) has mentioned that the presence of secondary vorticity on the piston face prevent the primary vortical structure from scaling in a self-similar way. This was observed from experimental studies conducted. The inviscid model proposed by Allen (1996) does not incorporate this secondary vorticity.

Experimental studies have been proposed to further study the validity of the simulations, on the existence of this secondary vortex. In the work of Allen (1996), dye was introduced into the flow at the junction of the moving piston and the stationary wall. Lighthill (1963) has suggested that in incompressible uniform density flows, all vorticity is generated at solid boundaries. Thus, if dye is introduced into the fluid at the location where vorticity is generated, the vorticity would be marked by the dye until viscous diffusion causes the vorticity to be diffused away from the dye.

By introducing dye carefully on both the duct wall and on the piston face a more accurate description of the mechanisms involved in the vortex roll-up may be achieved.

ACKOWLEDGEMENTS

Financial support of the ARC for this research is greatly appreciated.

REFERENCES

Allen, J. J.,1996, "The Formation of Vortex Sheets Close to the Junction of Moving Surfaces", *Univ. of Melbourne*, Ph.D Thesis

Burggraf, O. R.,1966, "Analytical and Numerical Studies of the Structure of Steady Separated Flows", *J. Fluid Mech.*, **24**, p.113

Chorin, A.,1969, "On the Convergence of Discrete Approximations to the Navier-Stokes Equations", *Math. Comput.*, **23**, p.341

Harlow, F. H., Welch, J. E.,1965, "Numerical Calculation of Time-Dependent Viscous Incompressible Flow of Fluid with Free Surface", *Phys. Fluids*, **8**, p.2182

Kim, J., Moin, P.,1985, "Application of a Fractional-Step Method to Incompressible Navier-Stokes Equations", *J. Comput. Phys.*, **59**, p.308

Lighthill, M. J.,1963, "Attachment and Separation in Three-Dimensional Flow" In *Laminar Boundary Layers*. Ed. L. Rosenhead. Oxford: Oxford University Press, p.72-82

Tabaczynski, R. J., Hoult, D. P., Keck, J. C.,1970, "High Reynolds Number Flow in a Moving Corner", *J. Fluid Mech.*, **42**, p.249